COMPUTATIONAL FLUID DYNAMICS FOR WIND ENGINEERING

R. PANNEER SELVAM

WILEY Blackwell

Computational Fluid Dynamics for Wind Engineering

Computational Fluid Dynamics for Wind Engineering

R. Panneer Selvam, Ph.D., P.E.
University Professor
Department of Civil Engineering
University of Arkansas, Fayetteville, AR, USA

WILEY Blackwell

This edition first published 2022
© 2022 John Wiley & Sons Ltd

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, except as permitted by law. Advice on how to obtain permission to reuse material from this title is available at http://www.wiley.com/go/permissions.

The right of R. Panneer Selvam to be identified as the author of this work has been asserted in accordance with law.

Registered Office(s)
111 River Street, Hoboken, NJ 07030, USA
The Atrium, Southern Gate, Chichester, West Sussex, PO19 8SQ, UK

Editorial Office
9600 Garsington Road, Oxford, OX4 2DQ, UK

For details of our global editorial offices, customer services, and more information about Wiley products visit us at www.wiley.com.

Wiley also publishes its books in a variety of electronic formats and by print-on-demand. Some content that appears in standard print versions of this book may not be available in other formats.

Limit of Liability/Disclaimer of Warranty
In view of ongoing research, equipment modifications, changes in governmental regulations, and the constant flow of information relating to the use of experimental reagents, equipment, and devices, the reader is urged to review and evaluate the information provided in the package insert or instructions for each chemical, piece of equipment, reagent, or device for, among other things, any changes in the instructions or indication of usage and for added warnings and precautions. While the publisher and authors have used their best efforts in preparing this work, they make no representations or warranties with respect to the accuracy or completeness of the contents of this work and specifically disclaim all warranties, including without limitation of any implied warranties of merchantability or fitness for a particular purpose. No warranty may be created or extended by sales representatives, written sales materials, or promotional statements for this work. The fact that an organization, website, or product is referred to in this work as a citation and/or potential source of further information does not mean that the publisher and authors endorse the information or services the organization, website, or product may provide or recommendations it may make. This work is sold with the understanding that the publisher is not engaged in rendering professional services. The advice and strategies contained herein may not be suitable for your situation. You should consult with a specialist where appropriate. Further, readers should be aware that websites listed in this work may have changed or disappeared between when this work was written and when it is read. Neither the publisher nor authors shall be liable for any loss of profit or any other commercial damages, including but not limited to special, incidental, consequential, or other damages.

Library of Congress Cataloging-in-Publication Data
Name: Selvam, R. Panneer, author.
Title: Computational fluid dynamics for wind engineering / R. Panneer Selvam.
Description: Hoboken, NJ : Wiley-Blackwell, 2022. | Includes bibliographical references and index.
Identifiers: LCCN 2022012496 (print) | LCCN 2022012497 (ebook) | ISBN 9781119845058 (cloth) | ISBN 9781119845065 (adobe pdf) | ISBN 9781119845072 (epub)
Subjects: LCSH: Wind-pressure. | Computational fluid dynamics.
Classification: LCC TA654.5 .S4395 2022 (print) | LCC TA654.5 (ebook) | DDC 624.1/75–dc23/eng/20220624
LC record available at https://lccn.loc.gov/2022012496
LC ebook record available at https://lccn.loc.gov/2022012497

Cover Design: Wiley
Cover Image: © HelloRF Zcool/Shutterstock.com

Set in 9.5/12.5pt STIXTwoText by Straive, Pondicherry, India

Contents

Preface

My computational fluid dynamics (CFD) for wind engineering journey started around January 1983 at Texas Tech University (TTU) when myself and Dr. Kishor Mehta were brainstorming on new research areas on a Saturday morning and what I can consider for my PhD topic. Before that, I did not know anything about CFD and not much in fluid mechanics except taking a four-semester course work in my undergraduate program. Sincse I had reasonable background on numerical methods and its application to solid mechanics from my master's work in India and in the United States, I decided to apply those concepts to wind engineering applications. Especially, the tornado force on building fascinated me because only after I came to Lubbock, TX, I came to know about tornado and its devastation. In India where I grew up, I was exposed to hurricane-type wind extensively, and this may be another reason for me getting into wind engineering research area. At that time, I did not realize what I was getting into. Dr. Mehta did say I might not realize my dream even after 80 years old. However, Dr. James McDonald (my advisor) and Dr. Kishor Mehta did support my idea, and I started to apply numerical methods in fluid mechanics for tornado forces on buildings. I did not do any substantial work in my PhD work, but it did open the CFD application for wind engineering research area. My next vertical advancement happened when I visited Commonwealth Scientific and Industrial Research Organization (CSIRO), Australia, as a research scientist to work under Dr. John Holmes during the summer of 1990. He is a fun and nice person to work with, and I am glad he gave me an opportunity to work on CFD application to thunderstorm downdraft modeling. There I met Dr. David Peterson, and he taught me the implementation of the SIMPLE method of solving the Navier–Stokes (NS) equations and law of the wall boundary condition. There I used CFD to compute velocity in a thunderstorm downdraft and flow over 3D building using k-ε turbulence model. The paper (Selvam and Holmes 1992) becomes the beginning of thunderstorm downdraft study in wind engineering. From there on different challenges in CFD for wind engineering were resolved and now we are in a much better situation for application in wind engineering.

Dr. Allan Larsen in 1998, Dr. Partha Sarkar in 2010, and Dr. Prem Krishna in 2002, 2008, and 2017 requested me to write review papers on CFD for wind engineering. Those experiences gave me chances to reflect and advance myself for further developments. In the recent years, Dr. Arindam Chowdhury from Florida International University has become another motivator to expand my journey. Dr. Chowdhury and his student Dr. M. Moravej provided me wind tunnel data for the 1:6 scale TTU building, and he explained to me the partial turbulence simulation (PTS) method reported in Mooneghi et al. (2016) paper. This helped me to learn more about turbulence effects on building and challenges in wind tunnel modeling. He is a great person, and he opened my mind to learn more about inflow turbulence generation methods and energy cascade in turbulence. This is a concept many did not apply in turbulence modeling. If this concept were understood for practical application, the CFD application would have progressed much quicker. Murakami et al. (1987) used recycling method of considering turbulence in the flow using large eddy simulation (LES) for the first time in wind engineering. The recycling method has been used in many applications for several years after that. I also tried to implement it and reported my findings in Selvam (1997), and I thought at that time the turbulence energy has to be maintained as time goes on. From the numerical experiment, I found that after some time, most of the turbulence energy got lost in the computation. This could be due to the numerical diffusion as well as the energy cascade phenomenon. Because of my ignorance, I did not report the details in any of my publications. In recent years, I learnt that because of energy cascade and 3D modeling, the energy from lower frequency is transferred to higher frequency and also the waves get stretched and twisted.

In this work, random Fourier-based inflow turbulence generation method is used as inflow in Chapter 5 and the peak pressure on building is computed. The program developed for this case can be used for building aerodynamics study without inflow and with inflow. This helps the student to learn the power of CFD to some extent. This tool also gives a chance for students to generate their own wind data and analyze them for wind spectrum. The other notable problem considered is the vortex shedding in 2D cylinders. This provides a pathway to understand the vortex-induced vibration (VIV) issues in thin structures and bridges. The program for that also is used for class instruction. The programs developed for this class can run on a personal computer, and this makes it easier for students to use. The outputs are written in a format suitable for tecplot visualization program. The open-source visualization programs like ParaView can be used, and it is not user-friendly. However, the data can be manipulated for other systems easily because the files are in ASCII format.

To perform CFD modeling for building and bridge aerodynamics, some understanding of the NS equation, properties of turbulence, turbulence modeling, introduction to finite difference method, and wind engineering is necessary, and they are introduced briefly in Chapters 1–4. At the end of each chapter, necessary homework problems by hand or computers are provided to have hands-on experience.

A brief review of CFD application in wind engineering is provided in Chapter 6. I do apologize to many researchers whose work I could not include in Chapter 6 due to lack of time and space. In Chapter 7, use of OpenFOAM for wind engineering is introduced.

This course material was developed in the summer of 2020 to teach in the fall semester. Before the Fall of 2020, I taught CFD class twice, which helped me to develop the course material more focused toward wind engineering. The material for the class was expanded as the courses were taught. I had few fresh graduate students like Ms. Kaley Collins, Mr. Caleb Chestnut, and Mr. Gerardo Aguilar who gave a lot of support to teach this class in addition to my graduate students (Mr. Sumit Verma and Ms. Zayuris Atencio). Because of them, I got Mr. Andrew Deschenes, Mr. Wesley Keys, and Mr. Yancy Schrader in my class as students. The participation of all of them really improved the course material.

Even though the course material is more toward wind engineering application, if someone wants to write their own program, numerical algorithms are provided and several programs are listed for their own development.

The course was taught in the Fall of 2020 with our own CFD research code and tecplot up to Chapter 5. The students ran the programs on personal computer, and that made it easier for students. The visualization program tecplot is user-friendly, but it is a commercial program. If someone wants to teach the Chapter 5 material using open-source CFD program OpenFOAM and open-source visualization program ParaView, they can do so by using the material in Chapter 7. The major challenge may be to adopt an inflow turbulence generator available from OpenFOAM.

Since no other textbook on computational wind engineering is available at this time, I developed a teaching philosophy after several months of reflection. If you have any comments for improvement after going over the material, please email it to me. This means a lot and I greatly appreciate. I do hope this material is useful for students, industry practitioners, and researchers. I would like to thank Dr. A. Chowdhury for going over the material and providing valuable comments for improvement. Finally, I like to acknowledge the financial support received from Airforce, Navy, NASA, NSF, FHWA, James T. Womble Professorship and the Department of Civil Engineering, University of Arkansas over the years to conduct many of the research work reported in this book.

References

Murakami, S., Mochida, A., and Hibi, K. (1987). Three-dimensional numerical simulation of airflow around a cubic model by means of large eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics* 25: 291–305.

Mooneghi, M.A., Irwin, P., and Chowdhury, A.G. (2016). Partial turbulence simulation method for predicting peak wind loads on small structures and building appurtenances. *Journal of Wind Engineering and Industrial Aerodynamics*. 157: 47–62.

Selvam, R.P. (1997). Computation of pressures on Texas Tech Building using large eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics* 67 & 68: 647–657.

Selvam, R.P. and Holmes, J.D. (1992). Numerical simulation of thunderstorm downdrafts. *Journal of Wind Engineering and Industrial Aerodynamics* 44: 2817–2825.

R. Panneer Selvam
University of Arkansas
September, 2021

1

Introduction

Fluid mechanics and heat transfer have extensive application. From aeronautical industry to automatic industry, it is applied to several areas. Some of the notable areas are:

1) Aeronautical industry – design of airplane to electronic devices
2) Automobile industry
3) Building and bridge aerodynamics (Selvam 2017)
4) Electronic cooling (Silk et al. 2008; Sarkar and Selvam 2009)
5) Environmental flow and heat transfer
6) Metrological flow and weather prediction
7) Hydraulic flow
8) Water treatment (Liu and Zhang 2019)
9) Wind energy

In all areas, computer modeling has been extensively used in the recent years, and this branch of computation is called computational fluid dynamics (CFD). CFD provides the detail of velocities, pressure, and temperature at every point at each time in the computational domain. This helps to create animation in time and provides the detail of the flow changes in time. To gather this much of information from experiment is very expensive. In certain situation like weather prediction, we cannot do any experiment and computer simulation is the only tool to predict the weather. The major challenge in CFD is to develop a reliable computer model for a particular application. If this is established for a particular application, it will be very useful in the design of the system.

The CFD is applied from single-phase flow to multiphase flow. In the multiphase flow, it can be liquid–vapor flow, solid–liquid flow, and solid–liquid–vapor flow. In these flows, chemical reactions can occur. Some of the challenging flows I encountered in the past 30 years are

Wind–bridge interaction: Here the bridge moves due to wind and hence beyond certain velocities the bridge can flutter as reported in Selvam et al. (2002). Below the critical velocity for flutter, the bridge will not have unlimited oscillations. The concept of moving grid has to be used in addition to regular CFD modeling. The Tacoma Narrow Bridge failed due to flutter for a velocity of 64 km/h (17.8 m/s) in 1940. The critical velocity for flutter for Great Belt East Bridge is 252 km/h (70 m/s) as reported in Selvam et al. (2002). The critical velocity

Computational Fluid Dynamics for Wind Engineering, First Edition. R. Panneer Selvam.

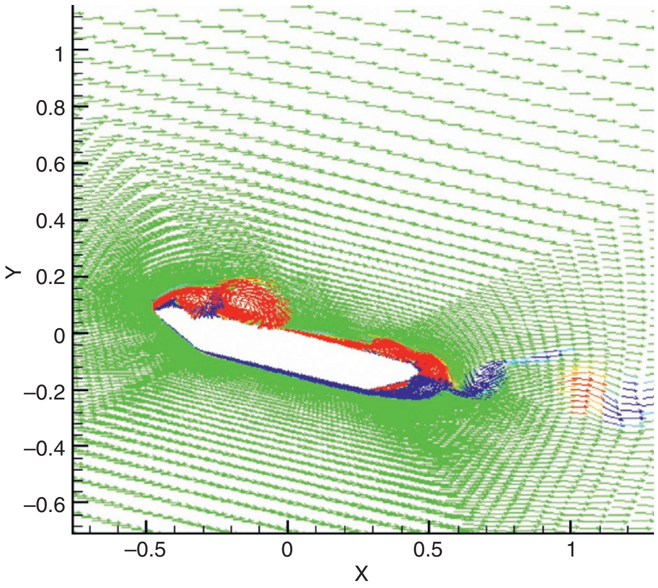

Figure 1.1 Flow around great Belt East Bridge during flutter condition.

depends upon the shape and structural properties of the bridge. The flow features during flutter condition are shown in Figure 1.1.

Heat transfer mechanism in spray cooling: Here, when a liquid droplet impacts a hot plate with a bubble growing in a thin liquid film; heat is removed due to complex interaction of droplet impact and vapor bubble. This high heat removal phenomena are explained in Selvam et al. (2006). For this, multiphase flow modeling of liquid and vapor is considered. In Figure 1.2 the liquid and vapor phases before the droplet impacts a vapor bubble in a liquid film are shown.

Tornado–building interaction: This study is reported in Selvam and Millett (2003, 2005). Here in a tornadic flow how a roof of a building is lifted up is explained using CFD. Figure 1.3 shows the velocity vector over the roof when a tornado-like vortex coincides with the center of a cubical building.

1.1 Brief Review of Steps in CFD Modeling

In the CFD modeling, the steps are very similar to well-established solid mechanics modeling. The major differences being most of the CFD applications are nonlinear and hence several iterations or time steps need to be performed.

Step 1: Grid Generation or Preprocessing: This may be the most time-consuming part if one has a complicated domain. If simple domain where in rectangular grid systems can be used, then the grid generation may be an easier task. Still one has to focus on the grid refinements in the boundary layer and in the regions of steep flow. Also, one has to make sure that grid spacing variation should not be high. The preferred ratio is 1.0–1.5. Very large ratios

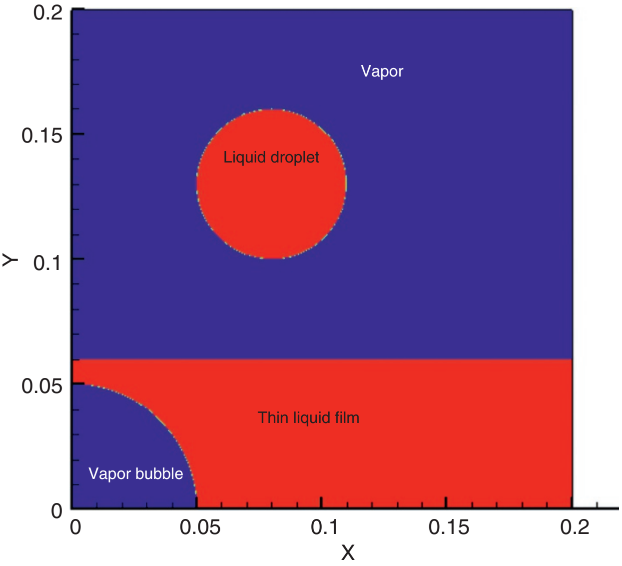

Figure 1.2 Multiphase flow modeling of liquid droplet impacting a vapor bubble in liquid film.

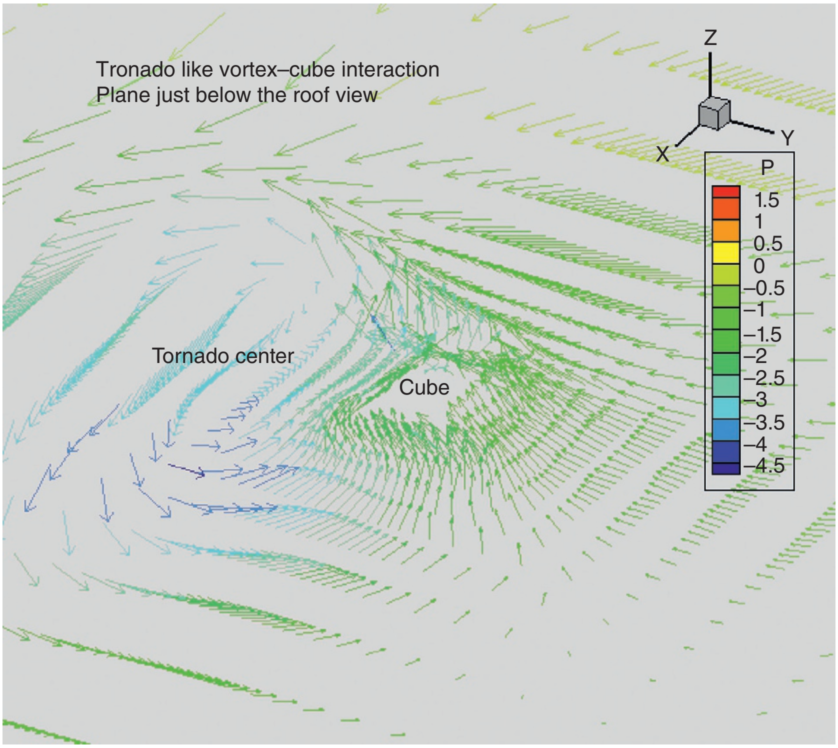

Figure 1.3 Velocity vectors around the roof when a tornado-like vortex coincides with the center of a cubical building.

like more than 5 or 10 are not preferable. For this step, extensive grid generation programs were developed in the recent years.

Step 2: Flow Solver: Once the grid is generated for a particular problem and the proper initial and boundary conditions are given for the problem, one can solve the Navier–Stokes (NS) equations. This is the most computer time-intensive step. For this several methods from direct to iterative procedures are developed to solve the $Ax = b$ equations. To reduce computer time, high performance or parallel computing is also utilized. Sarkar and Selvam (2009) utilized parallel computing to reduce the computer time from 50 to 3 days for spray cooling applications. They also compared the performance of different iterative solvers in the parallel computing environment.

Step 3: Postprocessing: In this step, the output from flow solver is processed to mine valuable information. Here this can be done by regular x–y graphs, contours, vector plots, and the combination of all. If the data is written for several time steps for the whole region, one can make animation using software like TECPLOT, and flow features can be investigated. The flow visualization technique is very sophisticated and some time it is an art than science.

If it is a design, then one can change the parameters of the flow variable or computational domain and further computer runs can be made for further investigations.

Benefits of CFD:

1) Data available for all points in space and time.
2) Inexpensive comparing to experiment. Especially with the developments in computer speed and memory, CFD programs can run in a personal computer. The major hurdle is validating the CFD with experiment to have reliability.
3) Visualization and animation of data to understand the physical problem is easy to implement. This helps anyone to understand complex fluid phenomena.

1.2 CFD for Wind Engineering or Computational Wind Engineering

In wind engineering, the loads on building and bridges are obtained from wind tunnel (WT) measurements or field measurements. The field measurement is very expensive and only very limited field studies are conducted like Texas Tech University building. Currently, WT is the major tool used to investigate forces on buildings and to develop code regulations like ASCE 7-16. In recent years, CFD is emerging as an alternate tool. For more than 30 years, different researchers raised its capabilities and slowly it is becoming a reasonable tool to be used in wind engineering because of the availability of high-performance computers with large storage capacities. The work reported by Selvam (1992) took more than a day for one computer run. With the current computer capabilities, one can solve the same problem in few minutes. Hence, the speed increased may be more than 100 times in a single processor. With multiple processors, we can increase the speed at least 10 times. If the CFD model is well validated with experiments, then it becomes the most economical tool compared to experiments. The way finite element method (FEM) is used in solid mechanics area in the industry and research nowadays, the hope is someday CFD will be a tool in wind engineering. This book is a stepping stone to achieve the preceding objective.

To apply CFD in wind engineering, one needs to be familiar with the following topics:

1) Meteorology or atmospheric flow
2) Fluid mechanics
3) Turbulence
4) Random process or stochastic process
5) Numerical techniques like finite difference method (FDM) or FEM for fluid mechanics
6) Wind engineering
7) Visualization
8) Structural dynamics
9) Fluid–structure interaction
10) Water (wave-storm surge)–wind–structure interaction as in hurricane
11) Grid generation
12) Parallel computing

In the current work, we may not touch topics beyond point 7 in the preceding list because of lack of time. For the other topics, we will go into detail only what we use in our work. We use simple computational domain to reduce the difficulty of making proper grid. One can see the grid generation complexity in the wind–bridge interaction study, as shown in Figure 1.1. In the industry for complex 3D problem, one or two engineers may be spending two or three months to make a proper grid. Even in wind engineering, we only work on straight wind. We will not discuss much about the other types of winds (tornado and thunderstorm downdraft) due to lack of time. From 1960 onward, field observations and WT testing have been used to find pressures on building. Because CFD takes lots of computer time and memory, only recent years CFD application in wind engineering has emerged with more reliability.

In hurricane-type sever wind, in addition to wind effects on structures, water surge and wave effect produce enormous damage. This leads to multiphase flow (water and air) effect on buildings. Future application may involve water–wind effect on structures.

References

Liu, X. and Zhang, J. (2019). *Computational Fluid Dynamics: Applications in Water, Wastewater, and Storm Water Treatment*. ASCE Publication.

Sarkar, S. and Selvam, R.P. (2009). Direct numerical simulation of heat transfer in spray cooling through 3D multiphase flow modeling using parallel computing. *Journal of Heat Transfer* 131: 121007-1–121007-8.

Selvam, R.P. (1992). Computation of pressures on Texas Tech Building. *Journal of Wind Engineering and Industrial Aerodynamics* 43: 1619–1627.

Selvam, R.P. (2017). CFD as a tool for assessing wind loading. *The Bridge and Structural Engineer* 47 (4): 1–8. [Review paper-available as opensource].

Selvam, R.P. and Millett, P.C. (2003). Computer modeling of tornado forces on buildings. *Wind & Structures* 6: 209–220.

Selvam, R.P. and Millett, P.C. (2005). Large eddy simulation of the tornado-structure interaction to determine structural loadings. *Wind & Structures* 8: 49–60.

Selvam, R.P., Govindaswamy, S., and Bosch, H. (2002). Aeroelastic analysis of bridges using FEM and moving grids. *Wind & Structures* 5: 257–266.

Selvam, R.P., Lin, L., and Ponnappan, R. (2006). Direct simulation of spray cooling: effect of vapor bubble growth and liquid droplet impact on heat transfer. *International Journal of Heat and Mass Transfer.* 49: 4265–4278.

Silk, E.A., Golliher, E.L., and Selvam, R.P. (2008). Spray cooling heat transfer: technology overview and assessment of future challenges for micro-gravity application. *Energy Conversion and Management* 49: 453–468.

2

Introduction to Fluid Mechanics

Mathematical and Numerical Modeling

Any physical problem can be modeled by algebraic equations, differential equations (DEs), and partial differential equations (PDEs). This formulation of the physical problem into mathematical equation is called mathematical modeling. Using algebraic equations for mathematical modeling, the solution is achieved by simple one equation as in Pythagoras theorem or by simultaneous equations. The one degree of freedom structural dynamics equation is a DE problem, and two-dimensional solid mechanics problems discussed in theory of elasticity is a PDE example. For DE and PDE problems, there are analytical solutions in regular regions (square or rectangle) for linear problems. For example, the beam problem in structural analysis is a fourth-order DE. The simply supported beam with uniformly distributed load has closed-form solution. For complicated load or change in cross section, one may sought to numerical methods like moment area method or virtual work method. In the same way for 2D and 3D problems, numerical techniques like finite difference method (FDM) or finite element method (FEM) are used to reduce the DE and PDE to algebraic equations. There are problems such as quadratic equation is a nonlinear equation and special methods have to be sought to find solution. All of these issues are applicable for fluid mechanics problems, and we will discuss the mathematical modeling part in this chapter and the numerical technique part in the next chapter.

2.1 Navier–Stokes Equations

The Navier–Stokes (NS) equations are the basic governing equations for the fluid mechanics problems. Solving the equation provides the solution to all kinds of problems, and some of the practical problems are introduced in the first chapter. For simple problems, analytical approaches are used and that is what you would have used in your undergraduate work. It is very difficult to solve complex problems using analytical methods, and hence numerical technique is preferred. This text will attempt to solve the NS equations using numerical technique like control volume or finite difference procedure.

For the details of the derivation of the NS equations using basic scientific laws, one can refer to Anderson (1995), Versteeg and Malalassekera (2007), and Cengel and Cimbala (2006). Due to space limitations, only the complete compressible flow equation is reported here.

Computational Fluid Dynamics for Wind Engineering, First Edition. R. Panneer Selvam.

To apply the compressible flow equations to practical problems, one needs to have good exposure not only in fluid mechanics but also in heat transfer. For basic understanding of heat transfer issues, one can refer to Incropera and DeWitt (1996). For the most of the incompressible flows, basic fluid mechanics exposure may be sufficient. The equations are reported in conservative and nonconservative form. Conservation form is preferred in the control volume procedures. Nonconservative form is used in the FEMs. For further discussion on advantages and disadvantages in using the aforementioned forms, one can refer to Anderson (1995) and other works.

2.2 Governing Equations for Compressible Newtonian Flow

$$\begin{aligned}&\underline{\partial\Phi/\partial \mathrm{t}} + \partial(u\Phi)/\partial x + \partial(v\Phi)/\partial y + \partial(w\Phi)/\partial z - \underline{\partial(\Gamma\partial\Phi/\partial \mathrm{x})/\partial \mathrm{x}}\\ &\quad - \underline{\partial(\Gamma\partial\Phi/\partial \mathrm{y})/\partial \mathrm{y}} - \underline{\partial(\Gamma\partial\Phi/\partial \mathrm{z})/\partial \mathrm{z}} - S_\Phi\end{aligned} \tag{2.1}$$

Time + (convection)	– (diffusion)	– source		
	Variable Φ	Γ	Source S_Φ	
Mass or continuity	ρ	0	0	(2.2)
Momentum	ρU	μ	$-\partial p/\partial x + S_x$	(2.3)
	ρV	μ	$-\partial p/\partial y + S_y$	(2.4)
	ρW	μ	$-\partial p/\partial z + S_z$	(2.5)
Internal energy	$\rho C v T$	k	$-p\,\mathrm{div}\,\mathbf{u} + \varphi + Se$	(2.6)
Equation of state for perfect gas: $p = \rho RT$				(2.7)

Here:

$$Sx = -\partial(\mu\partial U/\partial x)/\partial x + \partial(\mu\,\partial V/\partial x)/\partial y + \partial(\mu\partial W/\partial x)/\partial z + \partial(\lambda\mathrm{div}\mathbf{u})/\partial x + fx$$

where $\lambda = -(2/3)\mu$

Similarly, S_y and S_z can be developed

$$\begin{aligned}\text{Dissipation function } \varphi \text{ due to viscous stress} = \mu\{&2\left[(\partial U/\partial x)^2 + (\partial V/\partial y)^2 + (\partial W/\partial z)^2\right]\\ &+ (\partial U/\partial y + \partial V/\partial x)^2 + (\partial U/\partial z + \partial W/\partial x)^2\\ &+ (\partial V/\partial z + \partial W/\partial y)^2\} + \lambda(\mathrm{div}\mathbf{U})^2\end{aligned}$$

Here, Se is the source term, **U** is the velocity in the vector notation, and speed of sound $c = \sqrt{}(\gamma RT)$.

When the flow is incompressible, the density ρ is constant and the governing equations for velocity and pressure are:

$$\text{Continuity: div }\mathbf{U} = \nabla.\mathbf{U} = \partial U/\partial x + \partial V/\partial y + \partial W/\partial z = 0$$

$$\text{Momentum: } \rho[\partial\mathbf{U}/\partial t + \mathbf{U}.\nabla\mathbf{U}] + \nabla p - \mu\,\nabla^2\mathbf{U} = 0$$

Momentum in the expanded form for 2D or in the x and y directions:

$$\partial U/\partial t + U\partial U/\partial x + V\partial U/\partial y + \partial p/\partial x - \partial(\nu\partial U/\partial x)/\partial x - \partial(\nu\partial U/\partial y)/\partial y = 0$$

$$\partial V/\partial t + U\partial V/\partial x + V\partial V/\partial y + \partial p/\partial y - \partial(\nu\partial V/\partial x)/\partial x - \partial(\nu\partial V/\partial y)/\partial y = 0$$